Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 39 (2012) 19 – 27

Procedia Engineering

www.elsevier.com/locate/procedia

XIIIth International Scientific and Engineering Conference "HERVICON-2011"

The Fluid Flow in the T-Junction. The Comparison of the Numerical Modeling and Piv Measurement

Jaroslav Štigler[a], Roman Klas[b], Michal Kotek[c], Václav Kopecký[d], a[*]

[a,b]*Brno University of Technology, Victor Kaplan Department of Fluid Engineering, Technická 2/2869, Brno 602 00, Czech Republic*
[c,d]*Technical University of Liberec, Faculty of mechatronics, informatics and interdisciplinary studies, Studentská 2, Liberec 461 17, Czech Republic*

Abstract

This paper deals with the fluid flow in the T-junction with the 90° angle of the adjacent branch and with the 50mm diameter of each branch. The flow is organized as a combining flow. It means that there are two inlet branches and only one outlet branch. There are two possible kinds of the flow combinations for this T-junction. The CFD calculations and the PIV measurements were done for both flow combinations. Its comparison is presented in this paper.

Keywords: Combining T-Junction; PIV measurements; CFD calculations.

1. Problem introduction

Many papers have been written about the fluid flow in the T-junction. The T-junction is used for the division or the combination of a fluid flow. The case of the flow combination is often used for mixing of two different phases.

It is necessary to define a mathematical model of the T-junction and its coefficients in case of the T-junction handling. The simplest T-junction mathematical model is based on the assumption that the pressures at the ends of pipes, forming the T-junction, are the same. This assumption is not true, obviously. But this model has still been used because it is very simple and handy. It is very easy to apply

* Corresponding author. Tel.: +420-54114-2329; fax: +420-54114-2347.
E-mail address: stigler@fme.vutbr.cz.

1877-7058 © 2012 Published by Elsevier Ltd. Open access under CC BY-NC-ND license.
doi:10.1016/j.proeng.2012.07.003

it in case of a branched pipe system. In this case it is not necessary to define any T-junction coefficients. But it is clear that there are some losses in the T-junction. Therefore the more sophisticated description of the T-junction had to be derived. One of them was published by Miller [9]. There were introduced two coefficients of T-junction. Each of them was related to the different outflow branch in case of the flow division. But these coefficients have no any physical meaning because the value one of them could be negative. It means they cannot be classified as loss coefficients. Another improved description of T-junction was presented in the paper presented by Oka [8]. There still were disputable assumptions as for example, axis-symmetrical pressure distribution in all pipes. Therefore a new mathematical model has been established and presented by Štigler [6] and [7]. This mathematical model of T-junction consists of three equations. First of them is the power equation, second one is the momentum equation and the third one is the continuity equation. These equations are based on the only assumption, that the pressures are measured at the cross-sections where the velocity profiles are stabilized. It means the cross-sections where the influence of disturbances caused by the T-junction can be neglected. There were introduced two coefficients in this mathematical model. First of them is the power coefficient ξ_P. It is a scalar quantity which is proportional to the hydraulic losses in the T-junction. The second one is the momentum coefficient. This coefficient is a vector quantity which is proportional to the force of the fluid acing on the T-junction. It means that both coefficients have particular physical meanings. Moreover both coefficients can be divided into two parts. One part is called geometry coefficient and it represents the only geometry influence and the second part is a friction coefficient which represents the influence of the friction in the straight pipe. The total coefficients can be evaluated by the experiment or by the CFD computations. The friction coefficient can be also evaluated by experiment, CFD computations or by the empirical formulas. The geometry coefficients, which we are looking for, are then independent of the Reynolds number and they are also independent of the distances of the cross-sections where the pressures are measured.

This mathematical model of T-junction can be also used for an unsteady fluid flow in the T-junction. This is discussed in Štigler [5]. Then this mathematical model was extended for an arbitrary angle of the adjacent branch. It was presented by Štigler [4].

The most easy and probably cheapest way for setting the geometry coefficients is the CFD computations. Some results of the CFD computations are presented in a Louda [3] and Štigler [5] and [7]. But it is necessary to verify these computations by the experiment. Some comparison of CFD computations and pressure measurements had been done in Štigler [2].

The goal of this paper is to introduce the comparison between PIV (Particle Velocity Measurement) measurements and CFD computations of fluid flow in the T-junction with 90° angle of adjacent branch for case of combining flow. The combining flow in the T-junction means that there are two inlet branches and only one outlet branch.

Some information about the preliminary PIV measurements is presented in Kotek [1]. This comparison of measurements and CFD computation is very important for a validation of CFD computation.

2. Numerical model description

The geometrical model of the T-junction was created in the pre-processor GAMBIT 2.2.30, Figure 1. The diameters of all branches were 50mm. The length of the straight pipe was 200 mm and the length of the adjacent branch was 100mm. Then all branches were extended to prevent ill posed boundary condition. The length of the extension was equal to 25.D, where D is the pipe diameter. Each branch was marked by the letters A, B and C. It is clear from the Figure 1.

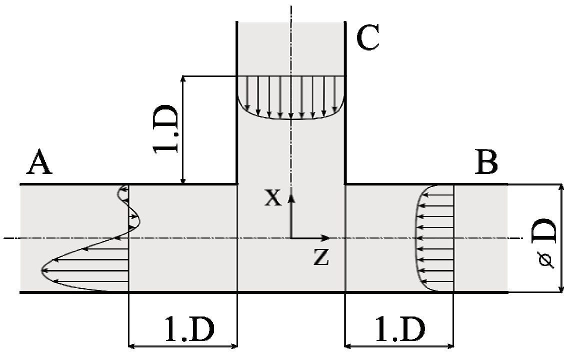


Fig. 1. T-junction

The boundary conditions were set on the corresponding surfaces. The condition *wall* (i.e. solid surface with defined surface roughness) was used for the pipe walls. The inlet and outlet cross-sections found in the cuts perpendicular to the main pipe axes were defined by boundary conditions mentioned in Table 1.

The computational mesh set in this way was consequently used for the analysis of the velocity and the pressure field in software Fluent 12.1. The setting of the solver was chosen according to the dominant flow i.e. unsteady. For other settings see Table 1.

Table 1. Mesh parameters and boundary conditions

Number of cells	approximately 2.3 million
Cell type	Hexahedral
The worst cell skewness	0.51
Material	liquid water
Turbulence model	k-ε realizable, non-equilibrium wall functions
Boundary conditions	flow rates ratios q_{ca}: (a) - outflow (b) – velocity inlet (c) – velocity inlet
	flow rates ratios q_{ac}: (a) – velocity inlet (b) – velocity inlet (c) – outflow

3. Experiment description

The PIV system was used for the experimental study of the flow in the T-junction. The model of the T-junction was made from the optical glass. The fluid flow measurement in the model with the circular cross-section of the branches meets many procedural problems such as image distortions, border refractive index changes and the image shift in the tube intersection. These problems emphasize during the image analyzing and flow velocity processing. Therefore the 3-cameras alignment was used to avoid these complications.

The investigated area was illuminated by the Nd:YAG 532 nm green pulse laser horizontally set to the model. The first camera was fixed above the model to observe the T-junction area. The next two cameras were put together and fixed in angle 45° with laser plane. Laser and cameras arrangement is sketched in Figure 2.

The test fluid was seeded with Rhodamine B coated 10μm particles. Rhodamin B absorbs the laser light of 532nm and emits the light of wave lengths close to 570nm. Orange optical filters (for wave length 570nm) were mounted on the lenses of all cameras to reduce the glass wall reflections.

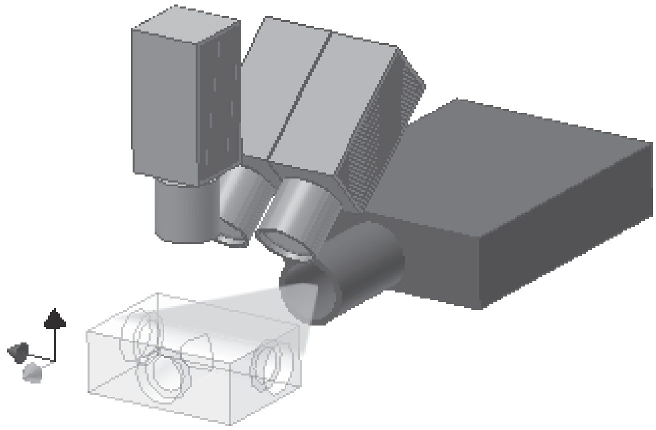


Fig. 2. Measurement system alignment

Three cameras captured the laser lighted plane with a different angle and non-equal magnitude. Image analyzing algorithms were designed to assemble these camera's images. The calibration procedure was split into four steps:

1. calibration images recording and pre-processing,
2. camera image dewarping,
3. dewarped image rotation and the change of magnitude,
4. images assembling.

The special calibration target was manufactured for the purpose of the calibration. The target filled whole area of the T-junction model. All camera images were calibrated and dewarped separately with appropriate calibration model and all images were assembled into one data set. Assembled image was analyzed with standard PIV correlation and validation procedures. The vector flow field, streamlines and velocity profiles were than calculated.

4. Results comparison and discussion

The four different flow orders are possible for this case of 90° T-junction. There are possible two flow orders for the flow division and two flow orders for the flow combination. These flow orders can be marked this way Div A, Div C, Com A, Com C. Div means the division and Com means combination. The letters A and C says in which branch the total flow rate is flowing through.

The results comparison was done for two cases of the flow orders Com A and Com C in this paper. It is necessary to define flow rates ratio for both cases. In case of Com A it is defined as $q_{(ca)}=Q_{(c)}/Q_{(a)}$. In case of Com C it is defined as $q_{(ac)}=Q_{(a)}/Q_{(c)}$. Where $Q_{(a)}$, $Q_{(b)}$ and $Q_{(c)}$ are the flow rates in each branch of the T-junction. The six different flow rates ratios were measured for each case. This is apparent from the Table 2. The range of these flow rates rations is <0,1>.

Table 2. List of measured cases

Com A	$q_{(ca)}$	0,000	0,205	0,400	0,607	0,805	1,000
	Label	00	02	04	06	08	10
Com C	$q_{(ac)}$	0,000	0,206	0,404	0,602	0,800	1,000
	Label	00	02	04	06	08	10

There were two things which were compared. The first one was the comparison of streamlines. It represents an overall view and the qualitative comparison of fluid flow in T-junction. The second one is the comparison of velocity profile at each branch of T-junction.

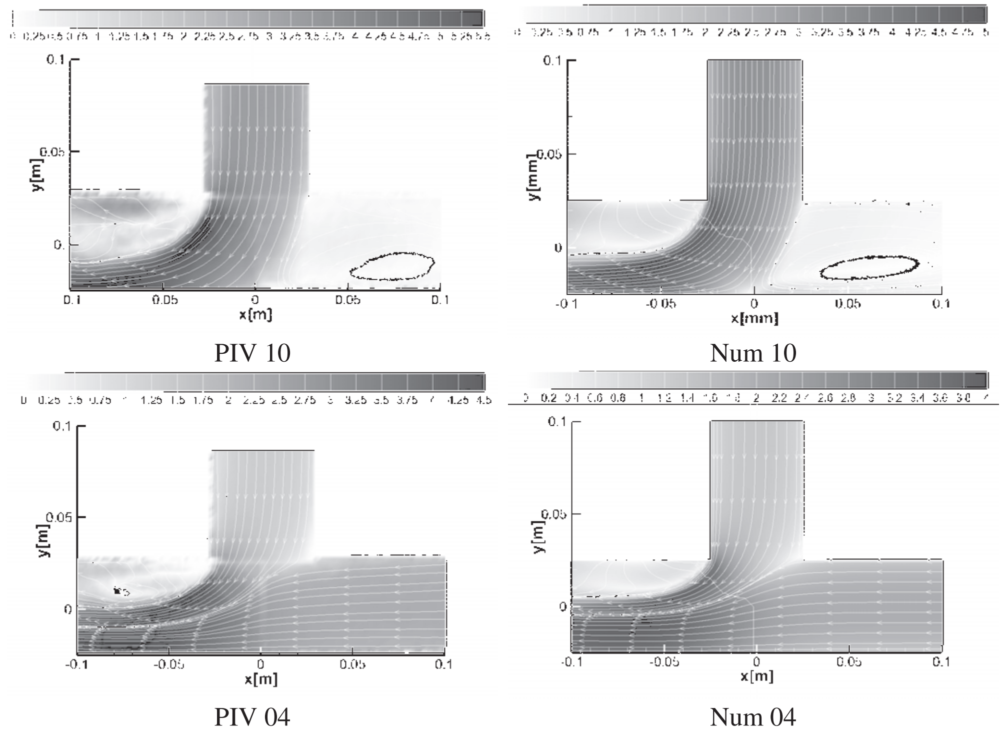


Fig. 3. Streamlines comparison for case of flow order Com A

The velocity profiles are taken in distance 1D from the border of the T-junction mixing area Figure 1.

The stream lines comparisons for case Com A are shown in the Figure 3. The stream lines comparisons for case Com C are shown in the Figure 4. There are showed only some cases mentioned in the Table 2, because of a little space in this paper. PIV results are in the left column and correspond numerical solution is in the right column in both pictures. It is apparent from the picture that the flow field got by PIV is very close to one which is got by numerical solution. The only think is that the maximal velocities are slightly bigger in case of PIV measurements of the case Com A.

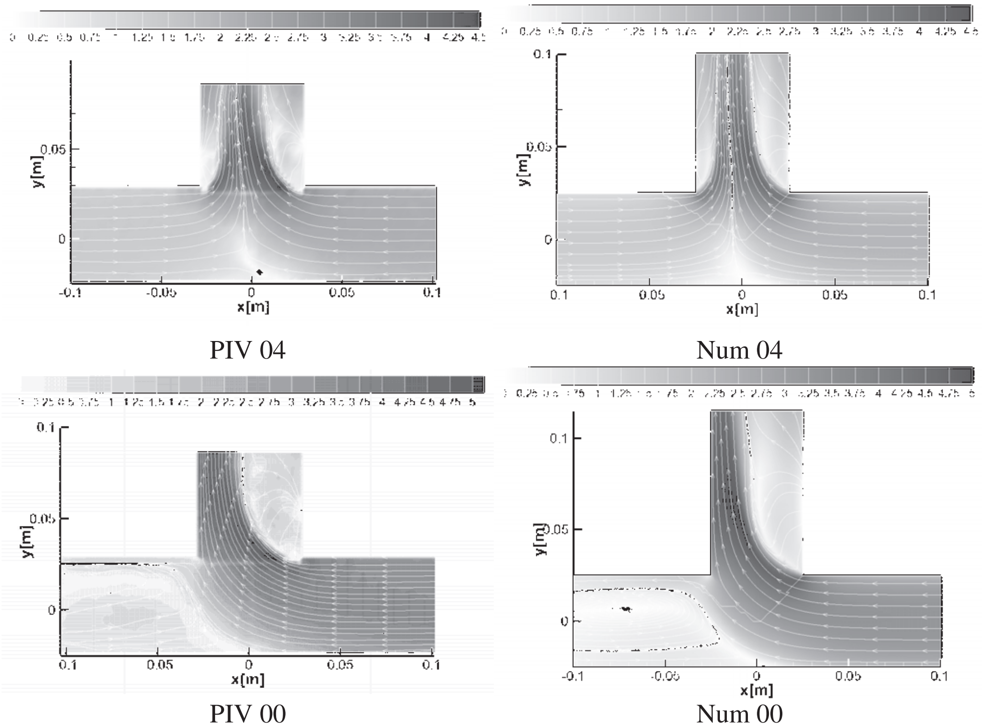
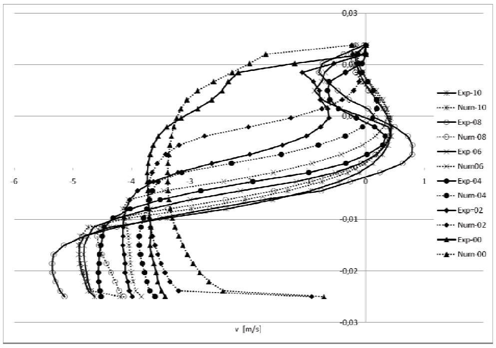


Fig. 4. Streamlines comparison for case of the flow order Com C

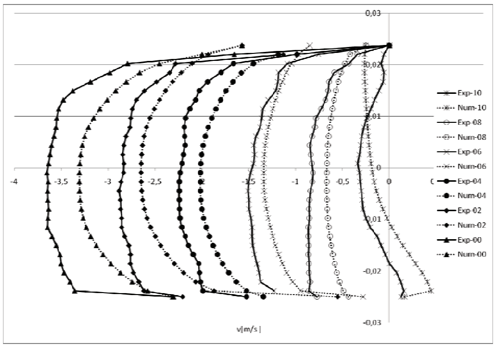
The velocity profiles are made from velocity components into the particular branch axis. The velocity profiles for the case of flow order Com A are drawn in the Figure 5 and for the case of the flow order Com C in the picture 6. The velocity profiles got by the numerical solution are drawn by the dot lines and the velocity profiles got by the PIV measurements are drawn by the solid line in both Figure 5 and Figure 6.

There are slight differences between the PIV measured values and numerical solution. It looks like that the numerical solution does not catch some deformation of velocity profile. These deformations are probably caused because the fluid flow in the branch C is preparing for the turn to the branch A and the part near the wall of the branch C is blocked by the flow. It represents the T junction influence of flow upstream in C branch. The axis velocity there is decreased more quickly and the regular velocity profiles are disturbed in that location. The velocity profiles got by the numerical solution are smoother than the PIV one. All this things are apparent for case of flow order Com A in the inlet branch C in Figure 5.

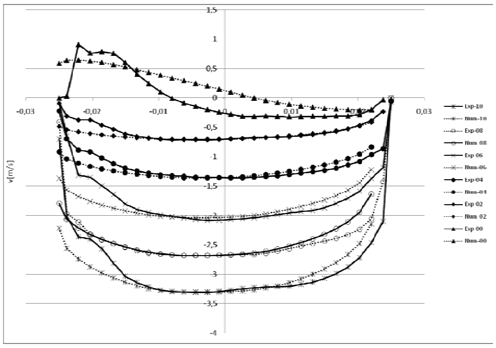
Also the velocities from numerical solution are smaller than the one got by PIV measurements. But it is necessary to take into consideration that these velocity profiles are taken only on the line across the branch and there is an unsteady fluid flow in the area of flow division. The comparison of the 3D velocity profiles over whole cross-section should have to be done for more detailed evaluation.



a) The outlet branch (a)



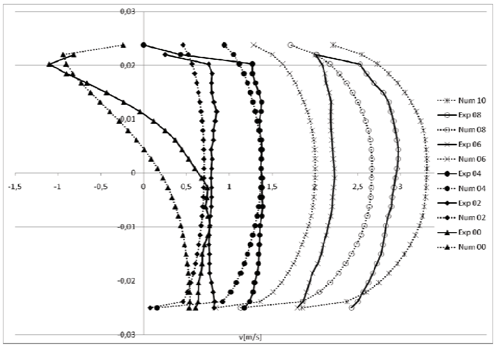
b) The inlet branch (b)



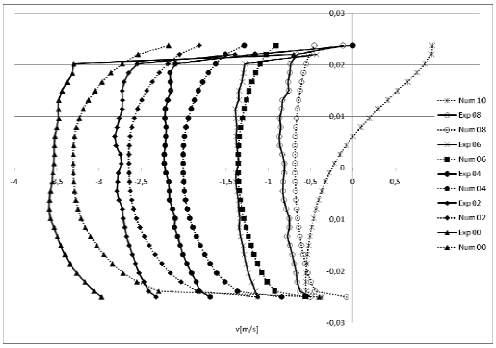
c) The inlet branch (c)

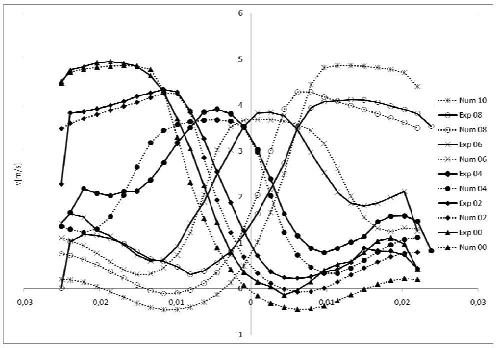
Fig. 5. The velocity profiles in distance of 1D from the border of T-junction for the flow order Com A

In was not possible to evaluate the data got by PIV for case Com C 10. Therefore the only velocity profile of numerical solution is presented in the Figure 6.



a) The inlet branch (a)





b) The inlet branch (b) c) The outlet branch (c)

Fig. 6. The velocity profiles in distance of 1D from the border of T-junction for the flow order Com C

It is possible to say at the end that the PIV measurements and numerical solution are very close. It means that it is possible to have confidence in the numerical solution.

Acknowledgement

The authors are grateful to the Grant Agency of Czech Republic (GACR) for funding this research under project "Mathematical and Numerical Modeling of Flow in Pipe Junction and its Comparison with Experiment". Registration number of this project is 101/09/1539.

References

[1] Kotek M., Jašíková D., Kopecký V.: *Experimental study of the flow field in T-junction model using PIV method.* In 17th international conference Engineering Mechanics 2011, Svratka Czech Republic, May 2011, pp. 303 – 306, ISBN 978-80-87012-33-8.

[2] Štigler J., Šperka O., Klas R.: *Mathematical model of the fluid flow in the Tee-Junction. The comparison of the CFD computation and the measurements.* In 17th international conference Engineering Mechanics 2011, Svratka Czech Republic, May 2011, pp. 607 – 610, ISBN 978-80-87012-33-8.

[3] Louda, P., Kozel, K., Příhoda, J., Beneš, L. Kopáček T.: *Numerical solution of incompressible flow fhrough branched channels*, Computers & Fluids, Vol. 46(1) 2010, p. 318-324, ISSN 0045-7930, DOI:10.1016/j.compfluid.2010.12.003.

[4] Štigler J.: *T-part as an Element of the pipe-line system. The introduction of the mathematical model of the fluid flow in T-part for the arbitrary angle of the adjacent branch.* In 25th IAHR Symposium on Hydrualic Machinery and Systems. September 20-24, 2010 Timisoara Romania. 2010 IOP Conf. Ser.: Earth Environ. Sci. 12 012102 (10pp).

[5] Štigler, J.: *Mathematical model of the unsteady fluid flow through tee-junction.* In 2nd IAHR International Meeting of the WorkGroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems. Scientific Buletin of the "Politehnica" University of Timisoara, Romania Transactions on Mechanics, Vol. 52, no. 6., 2007 pp.83-92. ISSN 1224-6077.

[6] Štigler. J.: *Tee junction as a pipeline net element. Part 1. A new mathematical model.* Journal of Mechanical Engineering. 2006, vol. 57, no. 5. pp. 249-262. ISSN: 0039-2472.

[7] Štigler, J.: *Tee junction as a pipeline net element. Part 2. Coefficients determination.* Journal of Mechanical Engineering. 2006, vol. 57, no. 5. pp. 263-270. ISSN: 0039-2472.

[8] Oka K., Ito H.: *Energy losses at tees with large area rations.* ASME Journal of Fluids Engineering, Vol. 127, January 2005, pp. 110-116, ISSN 0098-2202.

[9] Miller D. S. *Internal Flow. A Guide to Losses in Pipe and Duct Systems.* The British Hydromechanics Research Association, Cranfiled, Bedford, England, 1971, pp. 329.